EFFECT OF DIFFERENT TURBULENCE MODELS ON COMBUSTION CHAMBER PRESSURE USING COMPUTATIONAL FLUID DYNAMIC (CFD)

MOHD NASRUDDIN BIN AMIR NAZRI

A dissertation submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ABSTRACT

This thesis deals with the numerical study about the effect of different turbulent models on combustion chamber pressure during the event compression and combustion process using Computational Fluid Dynamic (CFD). The assessment is based on cylinder pressure and computational time predicted by the turbulence models. The vital point for the study is the on effect of different turbulence models on simulating the critical process of combustion in cylinder. The most accurate and time efficient models is k-ω-sst. The predicted results produce 58.2358 % discrepancy in term of cylinder pressure. The model also predicted the shortest convergence time which is 1573 minute. The selection of the models must be right in using numerical modelling approach in order to fulfil three major criteria which are accuracy, computational time and cost. This study consists of numerical modelling by using Mitsubishi magma 4G15 as baseline engine design. Engine speed at 2000 rpm was selected as baseline for initial condition. This project simulates the compression and combustion process right after intake valve closed until exhaust opened. For numerical modelling approach, there were six turbulence models selected which are *k-ϵ-standard, k-ϵ-RNG, k-ϵ-realizable, k-ω-standard, k-ω-SST,* and *RSM-Linear Pressure Strain*. The pressure data for turbulent models validate by compared to the experimental data. However, there are discrepancies of the results due to improper boundary condition and inherit limitation of model. For further simulation of combustion process must consider detail mixture properties, detail boundary condition and model extension for better accuracy.

ABSTRAK

Tesis ini berkaitan kajian berangka tentang kesan daripada pelbagai model aliran gelora dalam ruangan kebuk pembakaran semasa pemampatan dan proses pembakaran berlangsung dengan menggunakan kaedah dinamik aliran berkomputer, Computational Fluid Dynamic (CFD). Penilaian ini berdasarkan pada tekanan silinder dan masa pengiraan yang diramal oleh model aliran gelora. Perkara penting dalam kajian adalah untuk melihat perbezaan pada setiap model aliran gelora mengsimulasi proses pembakaran yang kritikal dalam silinder. Yang paling tepat dan waktu pengiraan yang cepat ialah k-ω-SST. Keputusan ramalan menghasilkan 58.2358% perbezaan tekanan silinder. Model ini juga meramalkan masa konvergen tersingkat iaitu 1573 minit. Pemilihan model haruslah tepat dalam menggunakan pendekatan model berangka untuk memenuhi tiga kriteria utama yang ketepatan, perhitungan waktu dan kos. Kajian ini terdiri daripada pemodelan berangka dengan menggunakan Mitsubishi Magma 4G15 sebagai dasar bentuk mesin. Kelajuan enjin pada 2000 rpm terpilih sebagai garis dasar untuk kondisi awal. Projek ini mensimulasikan proses mampatan dan pembakaran selepas injap masuk tertutup hingga injap ekzos tertutup. Untuk pendekatan pemodelan berangka, terdapat enam model aliran gelora dipilih model iaitu *k-ϵ-standart, k-ϵ-RNG, k-ϵ-realizable,k-ω-standart, k-ω-SST,* and *RSM-Linear Pressure Strain*. Data tekanan untuk semua model aliran gelora disahkan dengan dibandingkan dengan data eksperimen. Namun, ada perbezaan keputusan akibat dari keadaan sempadan yang tidak tepat dan keterbatasan model. Untuk simulasi masa hadapan bagi proses pembakaran, penelitian harus dipertimbangkan dari segi keadaan campuran, keadaan sempadan, dan model sambungan untuk ketepatan yang lebih baik.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

A & B	Empirical constant equal 4.0 & 0.5
$D_{i,m}$	Diffusion coefficient for species *ith* in the mixture
∂	Partial
δ_{ij}	Kronecker delta
ϵ	Epsilon
e	Specific total Energy
F_i	External body force from interaction with dispersed phase in *ith* direction
h	Sensible entalphy
h_j	$\int_{Tref}^{T} c_{P.J}$ Dt with T_{ref} = 298.15K
$J_{i,\, i}$	Diffusion flux of species *i*
k	Kinetic energy
k_{eff}	Effective conductivity
$\dot{m}$	Mass flow rate
m_j	Mass fraction of species *j*
ρ	Density

g_i	Garavitational body force
p	Static pressure
Ri	Net rate of production of species *i* by chemical reaction
S_h	Additional volumetric heat sources (example: heat of chemical reaction)
Si	Rate of creation by addition from the dispersed phase
t	Time
τ_{ij}	Stress tensor
μ	Fluid dynamic viscosity
u_i & u_j	The *ith* and *jth* cartesian component of instantaneous velocity
ω	Omega
Y_R	Mass fraction of a particular reactant R

LIST OF ABBREVIATIONS

CFD	Computational fluid dyamic
DNS	Direct numerical simulation
k-ϵ	K epsilon
k-ϵ -realizable	K epsilon realizable
k-ϵ-RNG	K epsilon renormalize group
k-ϵ-standart	K epsilon standard
k-ω	K omega
k-ω-SST	K omega shear stress transport
k-ω-standart	K omega standard
LES	Large eddy simulation
Pa	Pascal
RANS	Reynold averaging Navier-Stokes
RPM	Revolution per minutes
RSM	Reynold stress model
RSM-LPS	Reynold stress model linear pressure strain
TDR	Turbulence disipation rate
TKE	Turbulence kinetic energy
SDR	Specific dissipation rate

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Turbulence is that state of fluid motion which is characterized by apparently random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena. Turbulence can be seen in most cases in daily life such flow at buildings, cars, airplanes, fans, combustion chamber and many more. The successful of turbulence modeling increase in numerical simulation (Sodja, 2007). In these past years, many problems that involve turbulence flows are solve by using CFD for example fluid mixture, internal and external flows and in-cylinder flows. CFD approach provides user for gaining insight into in-cylinder flow (Payri et al., 2003). The view can be one of the result interpretations because the different is significant. The main importance of CFD approach is to attributes of both accurate and computationally fast to solution time (Kulvir et al., 2004). Hence, time consuming is crucial since the standard processor is just average rather that high capability processor that being used in high level or industry. However, that result should be acceptable in order to valid the CFD approach. After all, uncertainty of mathematically modeling turbulence is reflected in the large variety of models available (Kulvir et al., 2004). From here, the problem of choosing the right turbulence models due to right problems in terms of processing time and accuracy is important.

1.2 PROBLEM STATEMENT

From the findings, there are lots of turbulence models that available. But, the problem comes when selecting the right models for the right problems. Therefore, deciding the right turbulence models is not simple. The other concern is to reduce the amount of time that consume during the calculation process. So, the problems are to comparing turbulence models which is suit for in-cylinder flow and combustion study. Particularly, the purposes are to study the effect of turbulence models in term of accuracy to computational time.

1.3 OBJECTIVE

The objectives of this project are:

- To study the effect of different turbulence models on combustion pressure.
- To compare and validate each turbulence model's prediction with experimental data.

1.4 SCOPES

The scope of study covered the study and analysis on the effects of turbulent models and the accuracy due to processing time. Details scopes of this project consist of the following:

- To simulate in-cylinder flow using CFD approach during both valves closed.
- Develop the 2D pent-roof and combustion chamber model based on Mitsubishi Magma 4G15 engine dimension.
- Grid generation and boundary condition setup.
- Simulation of several turbulent models.
- Validate CFD approach by compare pressure data with experimental data.

1.5 ORGANIZATION OF THESIS

This thesis consists of five main chapter, introduction, literature review, methodology, result and discussion and the last part is conclusion and recommendation. For Chapter 1 presents some findings that lead to problems statement, objective, scopes and flow chart of work. Chapter 2 is literatures that related to the study and become basic of study framework. Chapter 3 presents the dimensioning work on Mitsubishi Magma 4G15 engine, development of 2D model and generation of computational domain. The pre-processing setup is presented in order to attain grid generation and imported to the solver to analyze. Chapter 4 addresses the validation of the predicted results against experimental results of the cylinder pressure. Chapter 5 presents the important findings of the study and recommendation for future study.

1.6 FLOW CHART

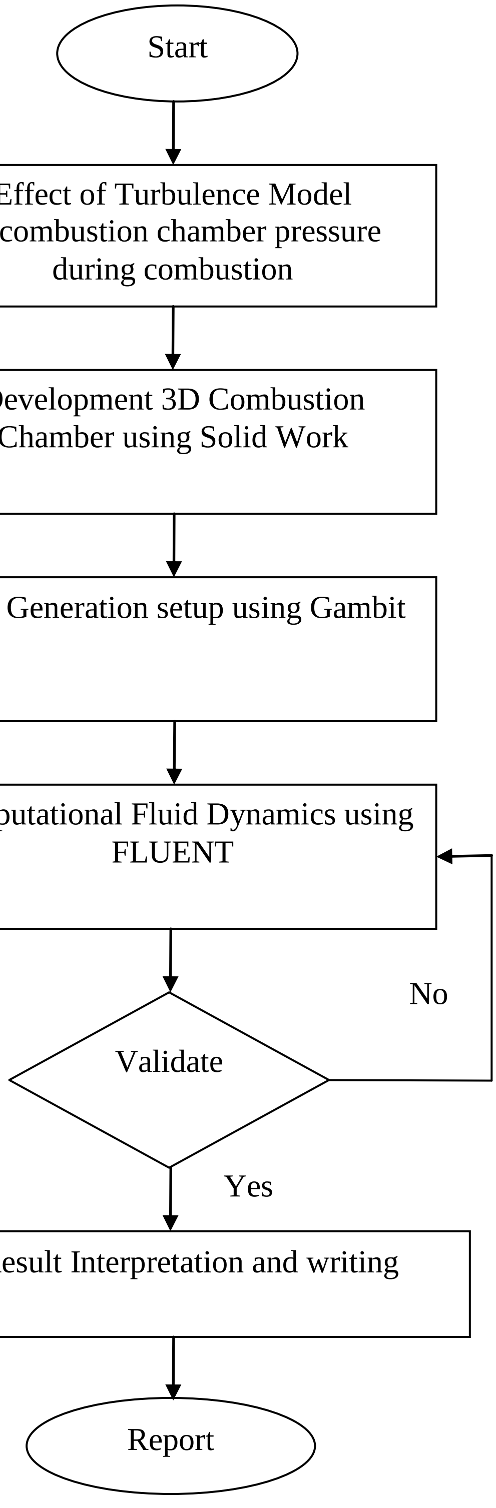

Figure 1.1: Project flow chart

1.7 SUMMARY

The purpose of the study is to acquire the main objective of the study related to the effects of different turbulence models. This chapter has summarized the titles, objective, scope, methodology, and the validation of study.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter deals with definition and characteristic of turbulence. Then, this chapter continues with the application of turbulence flow in in-cylinder flow study and the importance of the study about turbulence model for in-cylinder flow. Lastly, discussions continue with CFD approach for in-cylinder flow modeling and the advantages of CFD modeling for in-cylinder flow study.

2.2 TURBULENCE FLOW

In around 1500, Leonardo Da Vinci once thought about turbulence and draw called “La Turbulenza”. Leonardo describe turbulence as “Observe the motion of the surface of the water, which resembles that of hair, which has two motions, of which one is caused by the weight of the hair, the other by the direction of the curls; thus the water has eddying motions, one part of which is due to the principal current, the other to the random and reverse motion” (Ecke, 2005). So, it is understandable that turbulence has been long time studied and what has Leonardo quote is included in one of turbulence characteristics.

So, turbulence can be described as that state of fluid motion which is characterized by apparently random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena and results in increased energy dissipation, mixing, heat transfer, and drag (Sodja, 2007). If there is no three-dimensional vorticity, there is no real turbulence. There is no specific definition of turbulence model, but it has several characteristic features (Davidson, 2003), (Ziya, 2003), (Uygun et al., 2004) such as:

- Irregularity – As we all know, turbulence is random and chaotic. Turbulence flow is not constant respect to time. The flow consist of different scales of eddies sizes and fluctuate over time.

- Diffusivity – Turbulence flow increase in exchange the increment of momentum. As the turbulence flow increase, it will diffuse and become widely dispersed or spread out. The relation between resistances of friction to the diffusivity is vice versa. When one is increase, the other one is decrease.

- Large Reynolds Numbers – The basic knowledge that turbulence flow only happened only at high Reynolds number. Take fluid flow in pipes for example, transition happen at $Re \approx 2300$ and the turbulence flows start at $Re \approx 10000$.

- Three-Dimensional – This crucial characteristic is very important because turbulence flow is always three-dimensional. The flow is unpredictable and random. Even so, the equation is time averaging so that it can be solve easier.

- Dissipation – Turbulence flows are dissipative, which means the small (dissipative) eddies turns into internal energy. The smaller eddies receive the kinetic energy from larger eddies. The largest eddies get the energy from the main flow. This process that transfer the energy from main flow to the smallest eddies called cascade of energy as shown in Figure 2.1.

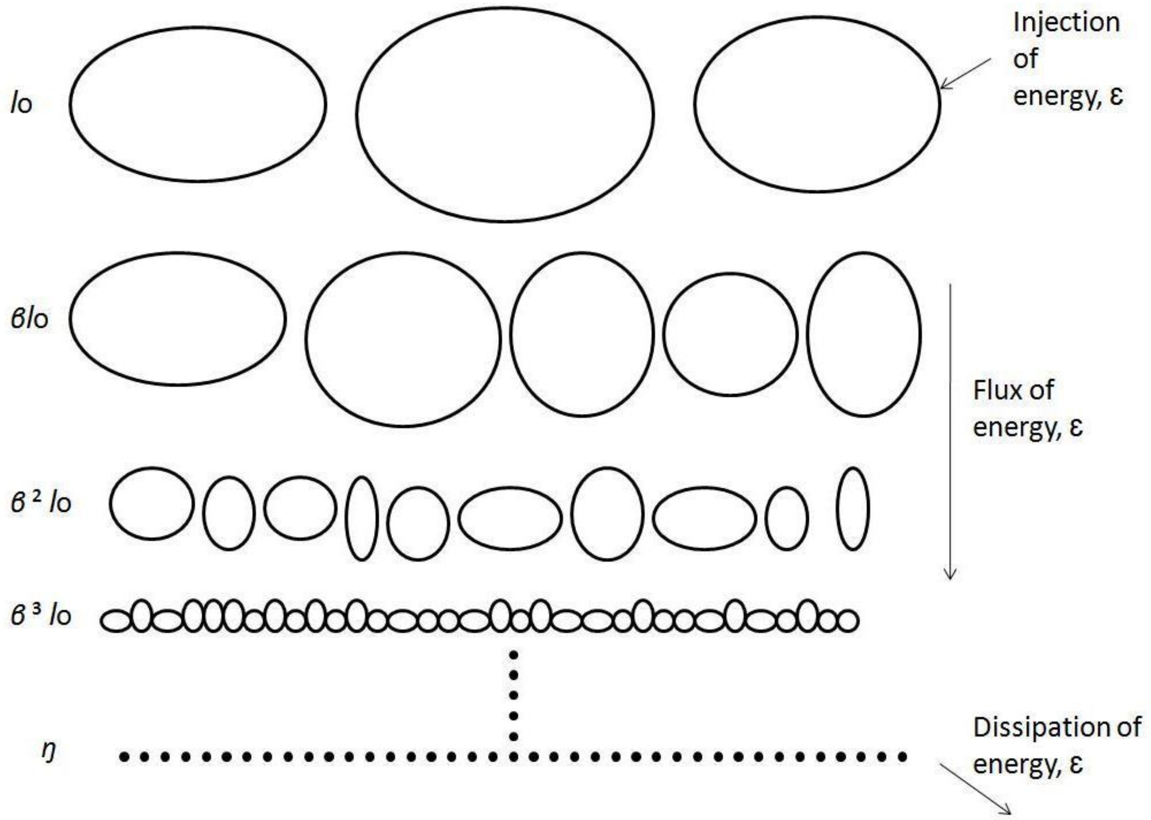

Figure 2.1: Energy cascade of turbulence.

Source: Ecke, (2005)

Since turbulence appears to most in our daily life, the effects of turbulence models are important since it is closer to nature and real cases. By the study the behavior of turbulence flows, the prediction of the desired result acquired by taking any precaution and initial awareness into study. This is important because in any cases such disasters, forecast and internal flow are amongst the need to predict in order to avoid such unwelcome accident. Industry and chemical process also involve fluid flows in packed beds (Gou et al., 2003). The distribution during the process is crucial to fulfill the criteria that demanded. It shows that the wide range of turbulence applications in the new era’s.

2.3 TURBULENT MODEL

The most efficient approach to solve turbulence flow is by modeling by based on numerical simulation. By this approach, all fluid motion can be resolve into prediction. Computational on turbulence models can be classified into several models.

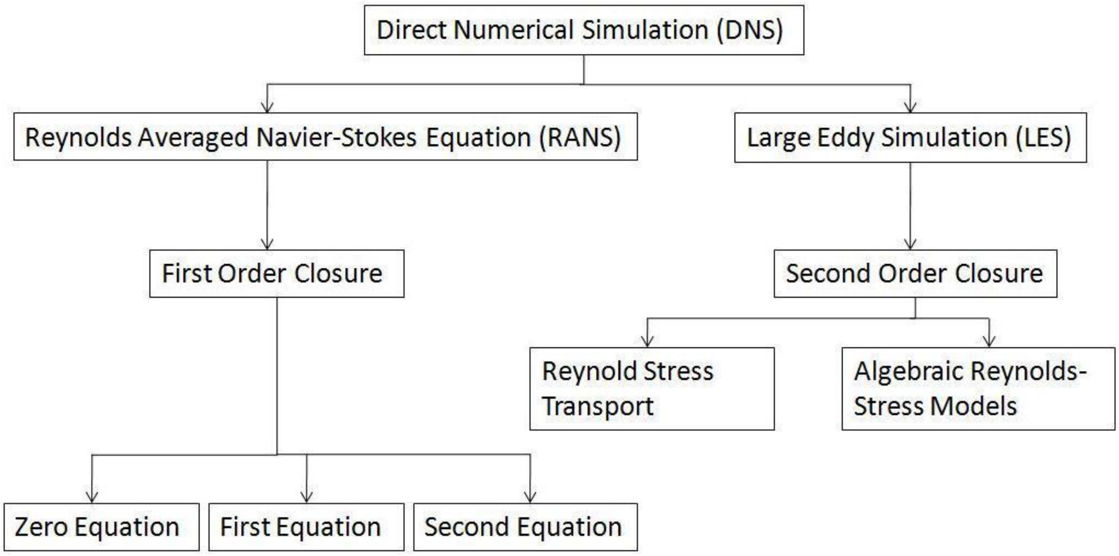

Figure 2.2: Turbulent models classification.

Source: Uygun et al., (2004)

As we can see from Figure 2.2, the turbulence models build from several classes. The classifications were made by previous researcher Uygun et al., (2004) based on result that computed, application, and complexity of the problems. From Figure 2.3, the simplest form of resolving turbulence is only solved the large eddies and modeled the effect of flux energy and dissipation of energy.

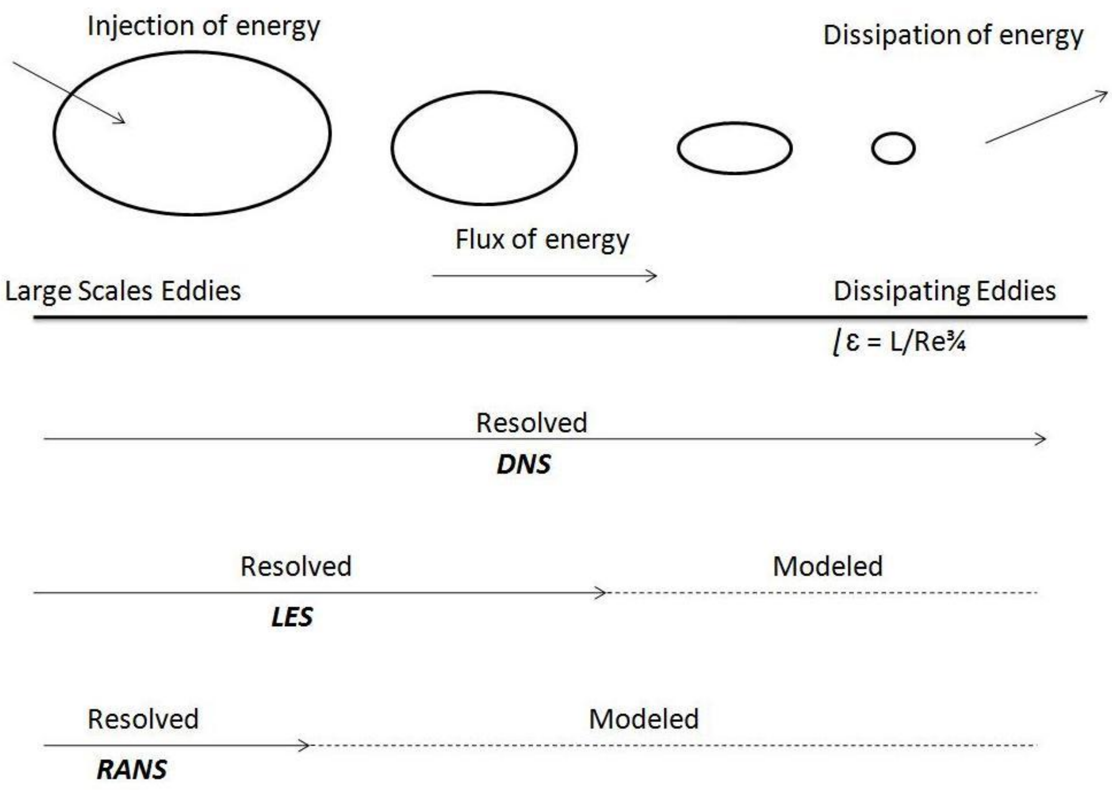

Figure 2.3: Extention to modeling for certain types of turbulence models.

Source: Sodja, (2007)

DNS is the most accurate method to solve turbulence flow (Uygun et al., 2004). This is because DNS does not need time averaging but solve the problem by numerical discretization. Hence all time and length scales are resolved. The solved problem is equivalent to those that attained by experimentally (Vengadesan and Nithiatasu, 2007). So, the accuracy level shown by DNS is idealized since the computed result is accurate as experiment. However, in order to capture all the turbulence scales, the computational domain must be as large as the physical domain or as large as the largest turbulence structure such eddy. It is important because to take into account every turbulence scales, the domain must be very fine grid. Usually, DNS used for simple geometries and to low Reynolds numbers (Vengadesan and Nithiatasu, 2007). From Figure 2.3, DNS solved all turbulence scales. Keeping in mind the relation the cost of a simulation goes up as processing time and grid size goes up. That is why DNS is so demanded method in term of cost and processor. Figure 2.4 show that the different eddies sizes under consideration during turbulence modeling.

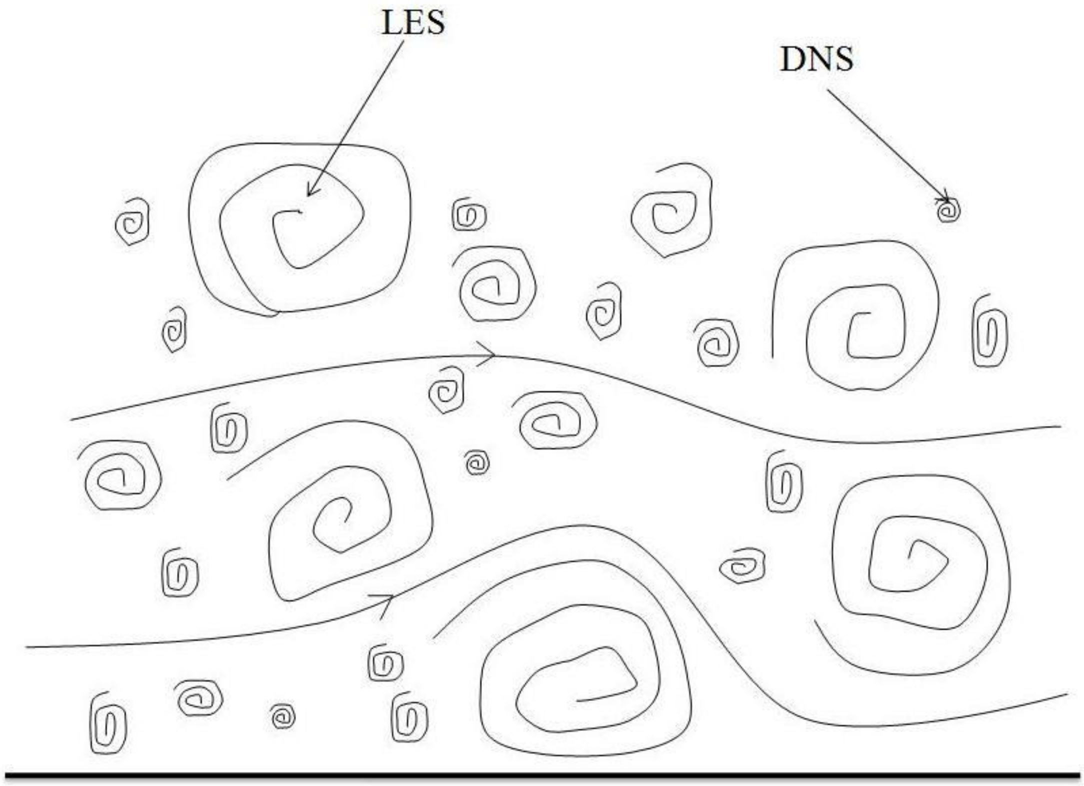

Figure 2.4: Large and small eddies.

Source: Uygun et al., (2004)

For LES, the observation based on large eddies that carries more energy then the smaller (Uygun et al., 2004). The subgrid-scale model used to simulate the energy transfer between the large eddies and the subgrid eddies (Uygun et al., 2004). The energy transport happen during cascade of energy process that continuum until the large eddies turns smaller eddies. That is why the size and energy make them effective for transportation of flow properties through interest. By referring to Figure 2.3, LES solve most of turbulence flow that consists of large scales and medium and modeled the small ones. After certain sizes of eddies, LES modeled the rest of turbulence flows. Even LES is considerable cheaper than DNS, LES still requires higher grid resolution in both the in order to solve the problems. By refer to Figure 2.4, LES solve only the large sizes of eddies that carries more energy, but DNS solve scales and size of all turbulence. That is why DNS far more accurate than LES but required higher cost and processing time.

Based on Figure 2.2, RANS can be divided into two main group, first order closure and second order closure. The discussion will follow those group and focusing on first-order closure.

- Algebraic models: These models contribute to the mixing length model in different ways and their models are the most popular amongst other algebraic models (Ziya, 2003). Examples of algebraic model are Cebeci-Smith model and Baldwin-Lomax model.

- One-equation models: Further improvement from previous models. There some interest in one-equation models of turbulence due to accuracy, simplicity of implementation and less demanding computational requirements (Ziya, 2003). Examples of one-equation model are Sparlat-Allmaras model and Baldwin-Barth model.

- Two equation models: The two–equation models have made truly significant contribution by introducing the famous k-e model. Then, Wilcox have pursued further development and presented successful application of k-e model (Ziya, 2003). Examples of two-equation model are k-ϵ and k-ω.

- Second order closure models: Right after the age of computer merge into new century, most improvements to model were abrupt these model shows some advantageous in sense that automatically accommodate complicating effect such streamlines curvature, rigid body rotation and body forces. However, because of large number of extra partial differential equations, complexity and computational cost is also increase as the demanding computer applicability (Ziya, 2003). Example of second order closure modes are Reynolds-Stress Transport and Algebraic Reynolds-Stress Models.

RANS models based in time-averaging of the dependent variables and the governing equations (Schluter et al., 2005).Technique solves the governing equation by modeling both large and small eddies, taking time-averaging of variables. From Figure 2.3, RANS is modeled the flows, that is why information supplied by these models is the time average of the variable and the fluctuating part. RANS is not represented directly by the numerical simulation, and are included only by means of turbulence models. These models have been extensively used for scientific and engineering calculations during the last decades. There are specially designed for high Reynolds numbers and distinguish separation of time scales related to the fluctuating behavior. Note that from Figure 2.3, the main advantages is the relative low computational cost involved compared DNS and LES since RANS mostly modeled the flows (Uygun et al., 2004). The bottle neck of these models is the difficulty to obtain highly accurate in addition to universally applicable models.

Nowadays, engineer and scientist are move towards to achieve the main objective to complete to the end the unsolved problems. Hence, the most accurate approach to turbulence simulation to directly the governing transport equations without undertaking any averaging or approximation other than the numerical discretization that performed (Tu et al., 2008). Through simulation, those turbulence flow that tested are solved by taking account some parameter to validate even so simulation is just a prediction.

From here, DNS show the most accurate method in CFD but highly cost and need very fine grid. So, LES is overtake by taking large eddies into account since large eddies carries massive energy. Even so LES is cheaper than DNS, but when compared to RANS reliability, LES is quite cost and demanding processor. So, LES modeling has problems with boundaries and is less computationally efficient than RANS techniques. RANS generally, k-ϵ especially is the most efficient in term of computational cost, time processing and processor demand. Even the result that obtained is not exactly same as DNS, but still acceptable and well known in engineering problems (Ziya, 2003).

2.4 TURBULENCE IN-CYLINDER FLOW

In-cylinder process model is simulating the full condition that in charge such thermodynamic cycle that containing spark ignition, turbulent flame propagation, heat dissipation, emission and knocking (Bi et al., 1994). Turbulence flow in-cylinder is important because variety of parameter that affect the consequences to the engine itself such emission, performance, durability, endurance and efficiency. Study showed that piston geometry is important in order to swirl the air-fuel mixture in combustion chamber (Hovart and Hovart, 2003). However, bowl shape plays significant roles near TDC and the early stages of expansion stroke by controlling ensemble-averaging mean and turbulence velocity (Payri et al., 2003).

During the intake stroke, air-fuel mixture is flowing through the intake manifold into combustion chamber. Relationships between flow structures within the runner and cylinder were seen to be strong during the intake stroke but less significant during compression (Justham et al., 2005). The in-cylinder flow diagnostics have been established in these few decades that provides greatly amount of information of flow and it is turbulence characteristics. By study and measure does improve combustion performance and help to understand engine performance. Researcher also noted that turbulence characteristic and intensity does make significant influence on combustion that is why accurate turbulence measurement is really important task (Kaneko et al., 1999).

From previous approach by researcher, turbulence model that used is RANS widely, followed by LES and DNS rarely. For RANS, k-ω model and k-ϵ model are used commonly since both gives inadequate result (Ogor et al., 2006). The requirement of processor to run RANS also lower and the running time is faster than LES and DNS this is another important key points why RANS used widely in CFD analysis (Sodja, 2007). Although RANS is faster and reliable, for high value and very important CFD analysis, DNS and LES usually used in order to achieve the accurate result that idealized for most engineering application (Venayagamoorthy et al., 2003). As far as studied carried on, the selection amongst turbulence model due to condition that went to